Frequency Optimization of Robot Support Structure for Inverted Operation

Santosh Haribhau Pawar, G. R. Selokar

Abstract*: : In this paper we have represented new method of optimization of the robot structure for various reverted operation In today's age the rapid development in robotics is been noticed, robots are developed for multipurpose operations in industrial field. The high inertia load are created due to emergency operation, the gradually applied and suddenly applied loads are developed, the structure should withstand this type of loads Furthermore, the robot has a certain frequency of operation, & the structure frequency needs to be higher than that of robot, to avoid failure due to resonance. The structure has a complex web of members hence finite element analysis is needed to perform analysis. The new structure, the robot is mounted in an inverted manner on the ceiling, this adds to the complexity of the analysis. The proposed method is compared with state of art systems.

***Keywords*: Finite Element Method, Modal analysis, Non Linear FEA, Robot Support structure design.**

I. INTRODUCTION

The robot birth took place in year 1920, by Czech Karel Capek by Rossum's Universal Robot, in Czech the robot stands for work. The term has been applied to a great variety of devices, such as humanoids (trying to mimic humans), domestic robots like robot vacuum cleaners and robot lawn movers, underwater vessels, military missiles, autonomous land rowers, etc. Almost anything that operates with a point of independence, usually under computer regulator, has at some point been called a robot. This paper deals with industrial robots, which consist of a mechanical arm with a number of joints, the robot arms are actuated by means of (electric) motor with the transmission lines. The controlling of the robot movement is done by computer controlled system this type of robot is usually called robot manipulator or simply manipulator. A widely accepted definition of a robot consistent with the Robot Institute of America (Spong and Vidyasagar, 1989) is: The robot is multi-functional manipulator and designed for the purpose of moving the materials, tools, an specialized device though repeated programmable device for different tasks. Industrial robots are essential in doing the complex works. They won't save the cost but increased the productivity and quality, and there is elimination of the labor work. The accuracy of the manipulator is based on mathematical model, and they should accurate. The model described the complicated non linear motion between robot motion and motor motion. The models are designed in such a way that they can withstand the robot structure load, and used in many fields like mechanical designed, performance simulation, control diagnosis, and supervision. A trend of light weight robot structure are now-a-days introduced, where the load is reduced in preserved payload capacity. This is motivated by cost reduction also as questions of safety, but leads to a weaker (more compliant) mechanical structure with enhanced elastic effects. For high performance, it's therefore necessary to possess models describing these elastic effects.

Revised Manuscript Received on March 30, 2020.

* Correspondence Author

Santosh Haribhau Pawar*, Ph.D., Scholar, Department of Mechanical Engineering, SSSUTMS, Sehore, (M.P.) India. Email: santoshpawar13@gmail.com

Dr. G. R. Selokar, Professor, SSSUTMS, Sehore, (M.P.) India.

II. LITERATURE SURVEY

Robert J. Sayer et al. (2003)[2] The finite element technique can be used for the dynamic analysis of existing equipment and also evaluate the dynamic characteristics of machine or structures prior to fabrication. FEA models can be used to find "stress stiffening" effects in rotating machine elements. This method is used to approximate the natural frequency and mode shapes of the mechanical systems. The FEM method is approximate numerical method. The accuracy of the solution is obtained by FEA depends upon the various factors. The factors are as follows:

1) Degree of refinement of the element mesh.
2) Appropriateness of the finite element types used to model a machine/structure.
3) Boundary conditions used at the limits of the finite element model.

Juren xie [2012] in his report FEA technology has became an important tool for evaluating the structural performance of offshore platforms. The evaluation is done when the structure is in special loading conditions such as earthquakes and vessel impacts. Although submerged oil wells were drilled from platforms as early as 1890's, the first modern offshore drilling/production platform was installed in 1947 off the coast of Louisiana at a depth of 6.1 m.

III. ROBOT SPECIFICATION

A. Specification

We are designing robot pedestal for six robot for given load condition in specification library.

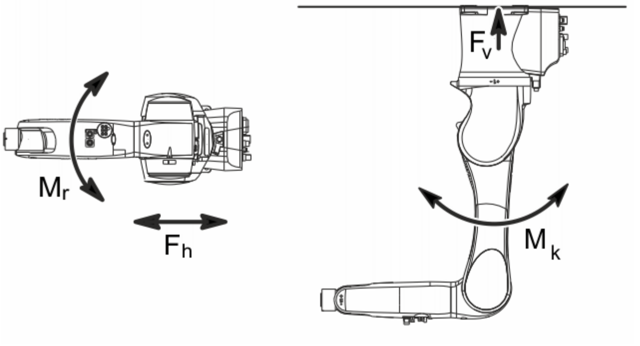

Fig 1. Load Condition

Table below estimates the load/torque/mass value at normal and maximum operating conditions.

Table- I: Load Condition robot kuka KR 6 R700

Type of load	Force/Moment/torque	
	Normal operation	Maximum load
vertical force	FV = 967 N	FV = 1297 N
horizontal force	FH = 1223 N	FH = 1362 N
tilting moment	Mk = 788 Nm	Mk = 1152 Nm
torque	Mr = 367 Nm	Mr = 880 Nm

Material- TATA YST 310

IV. BASIC STRUCTURE FINALIZATION

Table- II: HEX Dominant Meshing stress and Deformation

Element Size	Node	Total Deformation in (mm)	Stress MPa
15	19941	12.548	142.63
14.4	31500	12.548	163.54
10	40000	12.549	169.72
9.95	62477	12.554	172.71
9	68237	12.554	174.88
7.5	80000	12.555	176.39
7.49	115000	12.556	186.13

Table- III: Tet Element Meshing stress and Deformation

Element Size	Node	Total Deformation in mm	Stress MPa
30	24000	12.38	124.06
25	31326	12.371	133.26
17	40105	12.408	131.25
12.5	62000	12.748	157.63
11.5	72000	12.489	169.570
11.35	78000	12.513	160.74
9.5	109000	12.526	169.28

Table II and Table III shows the stress obtained in the structure, based on element size, we have observed at 40,000 node point the maximum load is been observed. By using path operation we have to find actual stress, it should not exceed than 125 MPA, for this purpose we have to select 10mm element size and Hexa dominant meshes.

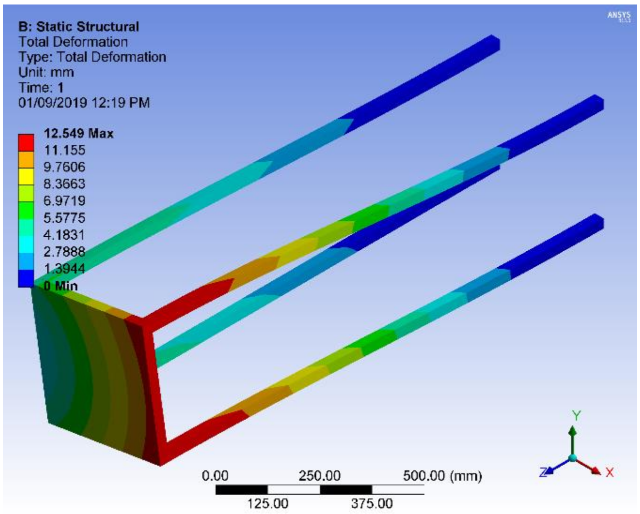

Figure 2: hex dominant meshing of 10 mm element size (total deformation)

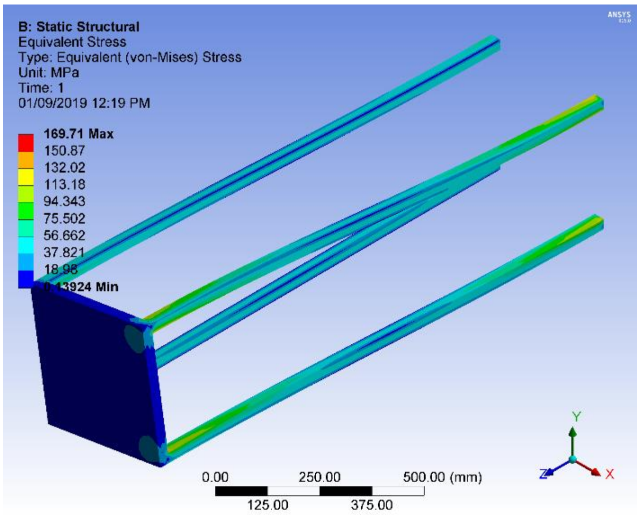

Figure 3: Hex dominant meshing of 10 mm element size (stress)

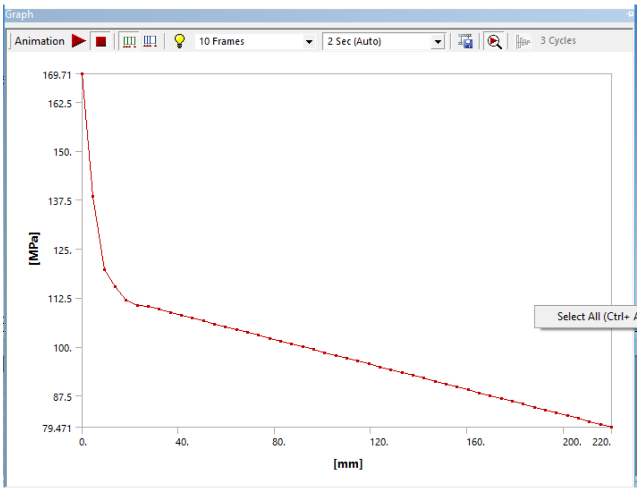

Figure 4: Path operation for 10mm hex dominant

B Non linearity

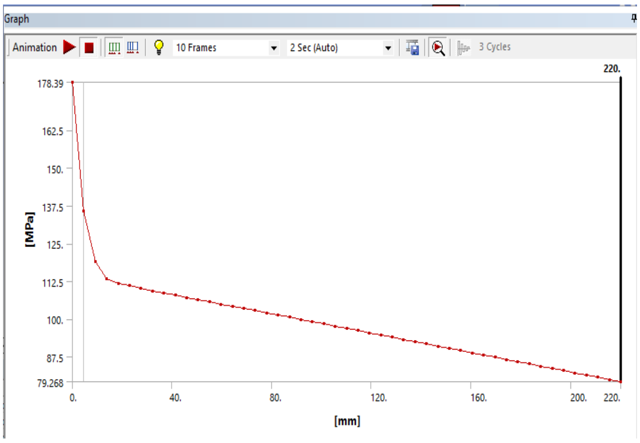

Figure 5: Geometric Non linearity checking

To opting the large deflection for geometric non-linearity we have to set sub steeping, and we get stress around 125 MPA. The Figure 3,4,5,6 demonstrates the values.

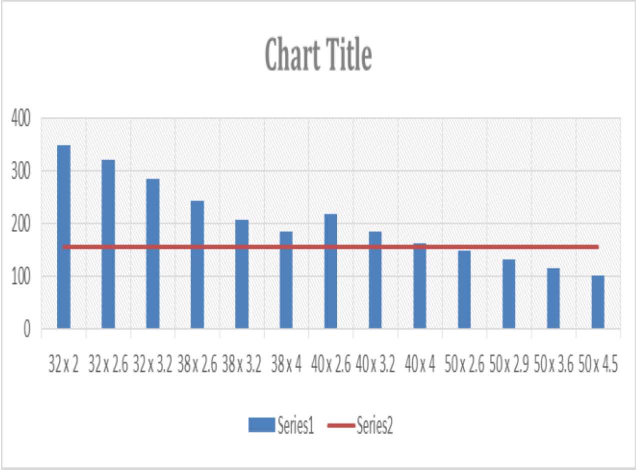

Figure 6: Graph of stress vs Leg size of TATA YST310 MATERIAL

Chart title in figure 7 shows us that Yield strength of the selected material is 310 MPA and assuming the safe FOS (factor of safety) as 2 selecting the leg size depending upon stress value, and model selected shouldn't exceed than 155 MPa. As we can see from the Graph 50X2.60, 50X2.90, 50X3.6, 50X4.50. is perfect so we are selecting this.

For finalization of load position and type of structure two cases are considered as follow.

Case-I: load condition as Following

Fx =1362N ,Fz =-1297N , My=1152 NM, Mz=880NM

Case-II : load condition as Following

Fy =1362N ,Fz =-1297N , Mx=1152 NM, Mz=880NM

Table IV: stress and Deformation of A type structure

	A TYPE			
	CASE 1		CASE2	
Leg size	Stress in MPa	Deformation in mm	Stress inMPa	Deformation in mm
50x2.6	59.293	0.40947	41.247	0.25339
50x2.9	44.765	0.37017	30.914	0.22917
50X3.6	38.282	0.30356	27.17	0.18839
50X4.5	31.292	0.24847	22.404	0.15464

Table V: stress and Deformation of V type structure

	V TYPE			
	CASE 1		CASE2	
Leg size	Stress in MPa	Deformation in mm	Stress inMPa	Deformation in mm
50X2.6	58.587	0.36352	38.143	0.20752
50X2.9	44.327	0.32819	29.936	0.18688
50X3.6	37.759	0.26867	26.076	0.15331
50X4.5	32.103	0.2193	22.308	0.12517

From above table it is clear that the maximum stress value obtained from case 1 so we are take force condition as mentioned in case 1.In both the structure the value of stress is optimized in v type structure so for next calculation we are selected the v type vertical structure. Now we required the maximum reach so we are design for L shape structure while the horizontal structure having fix horizontal length but varying vertical length from 400mm to 750mm. fig 8 and fig.9 shows that the V tape structure stress and Deformation analysis respectively.

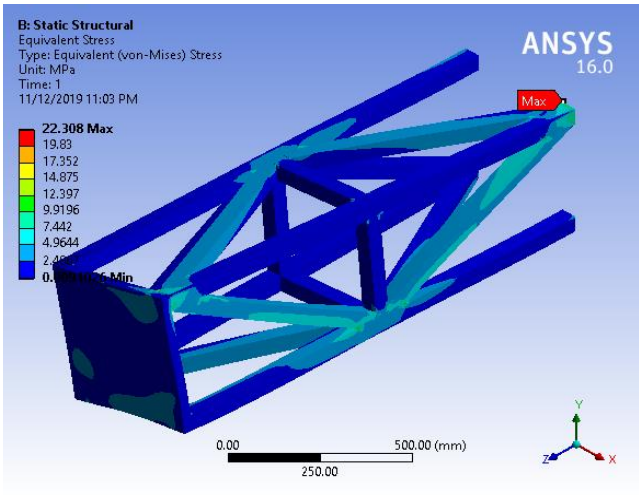

Figure 7: v type structure

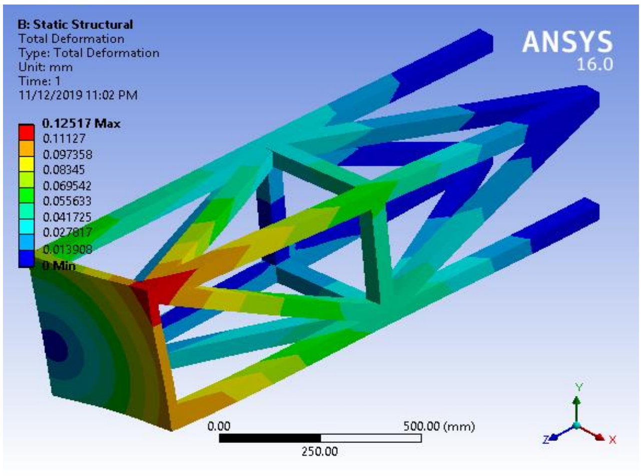

Figure 8: v type struture

Retrieval Number: E2775039520/2020©BEIESP
DOI: 10.35940/ijitee.E2775.039520
Journal Website: www.ijitee.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

V. RESULT AND DISCUSSION

Table VI Stress and deformation of horizontal v type structure

	50x2.6		50x2.9		50x3.6		50x4.5	
lenth	Def (mm)	Stress(N/mm²)	Def (mm)	Stress(N/mm²)	Def (mm)	Stress(N/mm²)	Def (mm)	Stress(N/mm²)
400	0.86998	61.853	0.78448	53.354	0.63942	42.024	0.5198	32.295
450	0.84349	59.654	0.76052	51.451	0.61955	40.468	0.5033	31.042
500	0.81321	56.94	0.733	49.092	0.59663	38.537	0.4841	29.483
550	0.77719	56.684	0.70042	50.827	0.56955	41.457	0.4611	32.303
600	0.74604	59.518	0.699	53.38	0.54251	43.462	0.4387	33.827
650	0.7072	57.41	0.63714	51.57	0.51739	41.823	0.6673	53.38
700	0.68693	50.777	0.6189	38.357	0.5028	32.347	0.6184	38.357
750	0.68297	51.172	0.61557	38.662	0.50011	32.629	0.4035	26.362

Table VI Stress and deformation of horizontal A type structure

	50x2.6		50x2.9		50x3.6		50x4.5	
lenth	Def (mm)	Stress(N/mm²)	Def (mm)	Stress(N/mm²)	Def (mm)	Stress(N/mm²)	Def (mm)	Stress(N/mm²)
400	0.87931	59.67	0.7929	51.52	0.6427	40.63	0.525	31.27
450	0.85201	57.49	0.7681	49.637	0.6258	39.09	0.508	30.027
500	0.82035	54.72	0.7399	47.228	0.602	38.6	0.489	30.242
550	0.78209	57.75	0.7048	51.858	0.5733	42.42	0.465	33.211
600	0.74316	60.49	0.6696	54.316	0.5442	44.3	0.441	34.625
650	0.70864	59.13	0.6384	53.159	0.5187	43.1	0.42	33.612
700	0.68869	50.64	0.6205	39.172	0.5043	32.27	0.46	35.736
750	0.686	51.02	0.6182	38.556	0.5025	32.55	0.407	27.441

Table VII: Modal analysis of A type structure

	50x2.6	50x2.9	50x3.6	50x4.5
lenth	Frequency(Hz)	Frequency(Hz)	Frequency(Hz)	Frequency(Hz)
400	20.180	21.187	22.990	23.750
450	21.497	22.142	23.341	24.506
500	21.892	22.547	23.759	24.941
550	22.386	23.052	24.286	25.498
600	22.975	23.661	24.941	26.199
650	23.456	24.158	25.485	26.786
700	23.711	24.423	25.773	25.877
750	23.706	24.418	25.764	27.078

Table VIII: Modal analysis of V type structure

	50x2.6	50x2.9	50x3.6	50x4.5
lenth	Frequency(Hz)	Frequency(Hz)	Frequency(Hz)	Frequency(Hz)
400	20.250	21.250	22.005	23.800
450	21.557	22.202	23.401	24.566
500	21.931	22.584	23.795	24.978
550	22.415	23.081	24.318	25.531
600	23.003	23.687	24.967	26.225
650	23.504	24.530	25.526	26.824
700	23.802	24.640	25.86	24.513
750	23.812	24.890	25.869	25.869

From the result obtained table, the data help for selection of horizontal A and V type structure. From the stress and frequency table, optimum values are selected for design purpose.

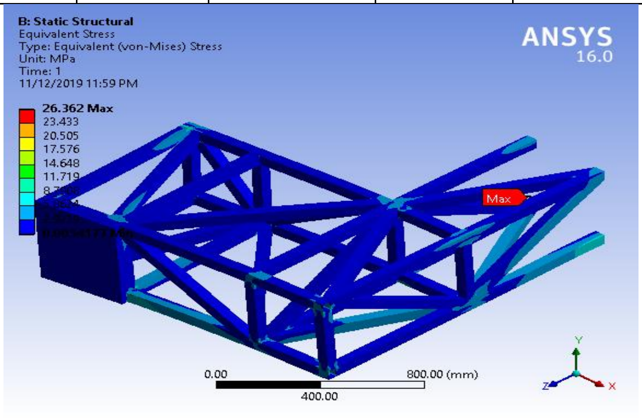

Figure 9: Horizontal V type Stress analysis

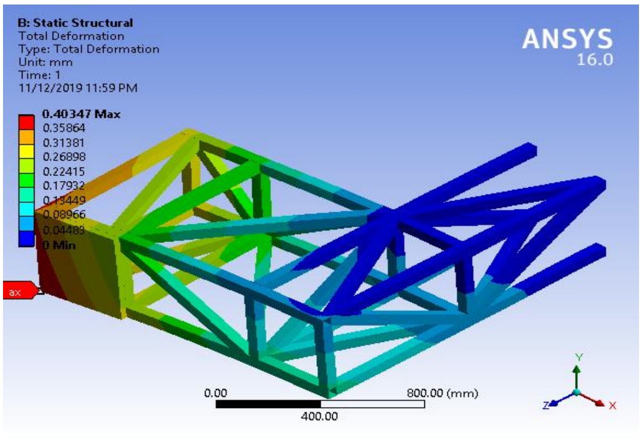

Figure 10: Horizontal V type deformation

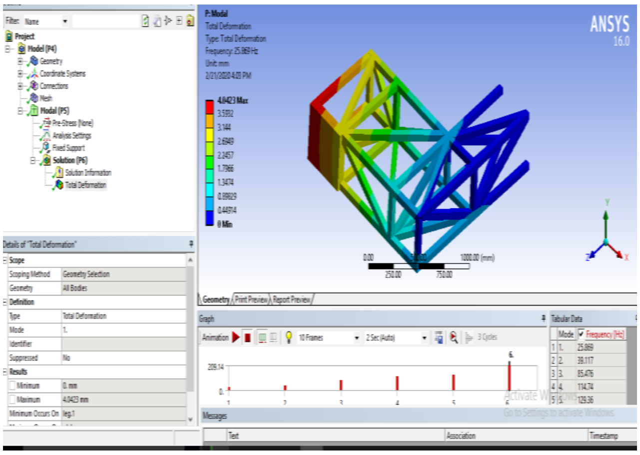

Figure 11: Horizontal V type modal analysis

For the finalizing robot structure we are provide a foundation and we are calculated a support reaction on each of the foundation.

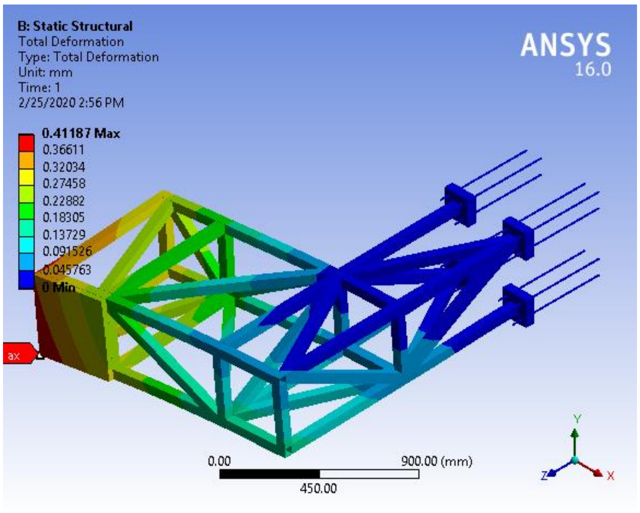

Figure 12: Horizontal V type with foundation deformation analysis

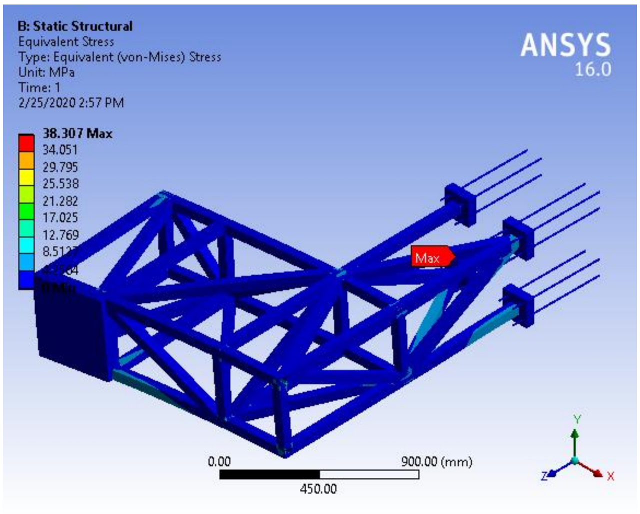

Figure 13: Horizontal V type with foundation stress analysis

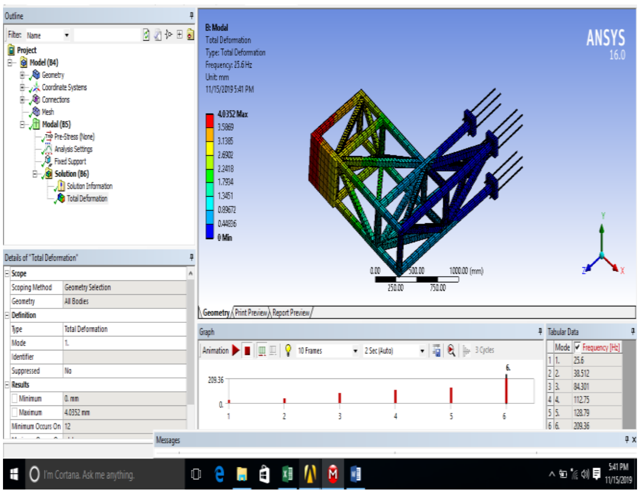

Figure 14: Horizontal V type with foundation modal analysis

VI. CONCLUSION

This robot support structure is optimized by considering all three parameter i.e stress, deformation and natural frequency. From the result obtained table, the data help for selection of horizontal A and V type structure. From the stress and frequency table, optimum values are selected for design purpose. From the analysis it is seen that horizontal V type structure is suitable than A type structure. The foundation connected structure having natural frequency 25.6Hz

REFERENCES

1. The Finite Element Modal Analysis of the Base of Welding Robot-Xiaoping Liao, Changliang Gong; 2010 3rd International Conference on Advanced Computer Theory and Engineering (ICACTE), 2010 IEEE.
2. Finite element analysis for structural performance of offshore platforms Juren Xie C FER technologies Canada; SIMULAI costumer conference,2012
3. Comparative Structural Analysis of 2-DOF Semi-Direct-Drive Linkages for Robot Arms; Jaydeep Roy, Louis L. Whitcomb; IEEE/ASME Transactions On Mechatronics, Vol. 4, No. 1, March 1999
4. A.H.Soni and M.H.F. Dado, Dynamic response analysis of 2-R robot with flexible joints. IEEE Conf. on Robotic and Automation (1987)
5. FEM Optimization for Robot Structure-Wang shijun, zhao jinjhan; IEEE ICIT'12,Bangkok, Thailand, 2002 IEEE.
6. Analysis and Optimization of Robot Gun Support Structure for Welding of Light Weight Vehicle Door Frame - Jatin H, ISSN (Print) : 2321-5747, Volume-3, Issue-6,2015
7. Analysis of the harvesting robot arm modal based on CAE, Jian Zhou, Zhimin Yang and Songlin Chen; Journal of Chemical and Pharmaceutical Research, 2014, 6(11):669-673

AUTHORS PROFILE

Santosh H. Pawar, is Phd Scholar in Department of Mechanical Engineering, SSSUTMS, Sehore, and. He has a total teaching experience of more than 7 years. His research interest includes nano composite abd FEA. He has contributed 4 research papers and also a Member of Institution of Engineers, Institute of Automobile Engineer, Indian society of Technical Education.

Dr. G. R. Selokar, Professor in the department of Mechanical Engineering of Sri Satya Sai University of Technology and Medical Sciences, Sehore is erudite professor, motivational teacher, eminent researcher and excellent institute builder with a dream to use science and technology for better community life. Prof. Selokar, an alumnus of Visvesvaraya National Institute of Technology (VNIT) Nagpur hold B.E. (Mechanical), M.Tech (Prod^{n} Engg.) and Ph. D. in Mechanical Engineering. In addition, he also earned PGDBM and MA (Pub. Ad^{n}) From RTM Nagpur University Nagpur in 1985 and 1995 respectively adding excellent blending of techno-managerial and administrative attributes.